Available online at www.sciencedirect.com
ScienceDirect
Energy Procedia 85 (2016) 539 – 549
Energy Procedia

Sustainable Solutions for Energy and Environment, EENVIRO - YRC 2015, 18-20 November 2015, Bucharest, Romania

Numerical evaluation of ventilation efficiency in underground metro rail transport systems

Catalin Teodosiu[a*], Viorel Ilie[a], Radu Dumitru[b], Raluca Teodosiu[a]

[a]Technical University of Civil Engineering, 122 – 124 Lacul Tei Bvd, Bucharest 020396, Romania
[b]Metroul S.A., 3 bis Gutenberg St, Bucharest 050027, Romania

Abstract

Ventilation system is the most important component of the subway systems when events involving heavy smoke occur. Consequently, the objective of this study is to analyse the ventilation efficiency in one of the most dangerous scenario: train on fire and stopped in the tunnel, the incident demanding immediate passenger evacuation. The ventilation strategy taken into account is based on mid-tunnel fan plant located in separate construction in conjunction with stations mechanical ventilation. The analysis is performed using Computational Fluid Dynamics (CFD) modeling. The approach is based on the introduction of source terms in conservation equations for energy and dioxide carbon (CO_2), in order to deal with the heat and CO_2 due to fire. The equation expressing the conservation of CO_2 is added to the basic equations governing a turbulent non-isothermal air flow in the CFD model. This method allowed achieving values of velocity, temperature and CO_2 concentrations all over the computational domain. The results show that the ventilation system taken into account provides the secure evacuation of passengers all over the simulation time. The evacuation process toward the nearest station is not at all disturbed by too high air velocities, high temperatures or dangerous CO_2 concentrations.

Peer-review under responsibility of the organizing committee EENVIRO 2015

Keywords: ventilation efficiency; indoor air quality; CFD – Computational Fluid Dynamics modeling; subway train fire

1. Introduction

Ventilation is one of the most important issue in underground metro rail transport systems. In addition, optimal functioning of the ventilation system is essential for emergency situations. In this context, it is worthwhile to mention that there were several important incidents in underground metro rail transport

* Corresponding author
E-mail address: cteodosiu@yahoo.com

Peer-review under responsibility of the organizing committee EENVIRO 2015
doi:10.1016/j.egypro.2015.12.241

systems around the world in the last years [1,2]. In these events, the main danger to passengers' life is the inhalation of smoke and toxic gases as these are released in almost complete enclosed areas [3].
On the other hand the investigation of ventilation efficiency is assessed nowadays by modern computational techniques. As a result, Computational Fluid Dynamics (CFD) approach is more and more used for studies dealing with ventilation strategies within different built environments (e.g. residential buildings, offices, hospitals, museums, etc.) [4]. In addition, CFD fire simulation has also become a common investigation method [5-7]. In line with this, more and more studies dealing with fire protection and life safety requirements in underground metro rail transport systems are based on CFD modeling. Lee et al. [2] simulated the effect of the smoke extraction system and fire shutters in subway stations in the case of fire from a kiosk located on the platform. Hu et al. [3] and Meng et al. [8] presented CFD investigation of most effective cooperative operation mode of the tunnel rail track area exhaust system and the platform ventilation system for the emergency scenario of a train on fire stopping beside the platform of a subway station. Chen et al. [9] investigated the effectiveness of the smoke control scheme in the Gong-Guan subway station (Taipei Rapid Transit System) under different locations of fires. Son and Chang [10] took into consideration for CFD modelling two of worst case scenarios (based on escape distance, escape time and fire zone) in the circumstances of a train which caught fire and stopped in the tunnel. Zhou and Zhang [11] conducted CFD simulations to improve the ventilation in the subway station fires, including the study of the air curtain effectiveness.

In this context, the aim of this study is to bring new elements related to CFD simulation in order to evaluate the ventilation efficiency in the most serious emergency scenario within underground metro rail transport systems: train caught fire and stopped in the tunnel, this situation demanding the traffic disrupting and passenger evacuation. The study investigates if the evacuation of the passengers toward the nearest station would be disturbed by too high air velocities, high temperatures or dangerous concentrations of toxic gases (expressed by CO_2 - carbon dioxide concentrations).

The ventilation system under investigation in this work is based on the following configuration: mid-tunnel fan plant in conjunction with mechanical ventilation systems provided in the stations of the underground metro rail transport system. As a result, the normal and emergency operation of this ventilation strategy are first presented in the case of two typical subway stations connected by two separate parallel tunnels. This is followed by their numerical description and results achieved in terms of velocities, temperatures and CO_2 concentrations.

2. Underground metro rail transport system configuration taken into consideration

The underground system taken into account includes over 88,000 m^3 and consists of two classic subway stations connected by two separate parallel tunnels, 1000 m long (see Fig. 1).

The two stations are the same, with two floors below the ground (Fig. 2). The lobby floor (3.4 m high and 13.9 m wide) has two parts, 32.5 m and 64 m long, respectively. The height of the platform floor is 4.5 m and the length is 146 m, while the width varies between 16.3 m (at the middle of the station) and 20.3 m (at the end of the station). The upper and lower levels are linked by three staircases. Each station has a total of four exits to outside ambient through large galleries communicating with the street (Fig. 2).

The ventilation system of this underground metro rail transport system is based on 6 air supply inlets (2 x 0.8 m^2) in each station. These inlets are positioned symmetrically on the width of the station, arranged in 3 pairs, at the edge of the upper level floor (Fig. 2). The air exhaust is occurring by means of the tunnel ventilation fan system (circular outlet of 7 m^2 in each tunnel, equipped with damper).

The subway train taken into account consists of six classic metro cars: 114 m long, 3.3 m wide and 4 m high (see Fig. 3). It is supposed that the subway train caught fire, because of technical incidents, and is stopped in one of the tunnels, at 250 m from one station and 750 m from the other one.

The ventilation system of the underground metro rail transport system operates as follows: in order to change the operating mode from normal mechanical ventilation to emergency (fire) ventilation, the supply air flow of the ventilation plants in the stations are brought to the appropriate fire situation (from 200,000 m^3/h to 400,000 m^3/h) while the exhaust air flow of tunnel ventilation fan system is changed from 200,000 m^3/h to 400,000 m^3/h; the damper of the tunnel ventilation fan system, communicating with the tunnel area where there is no fire, is closed (Fig. 4).

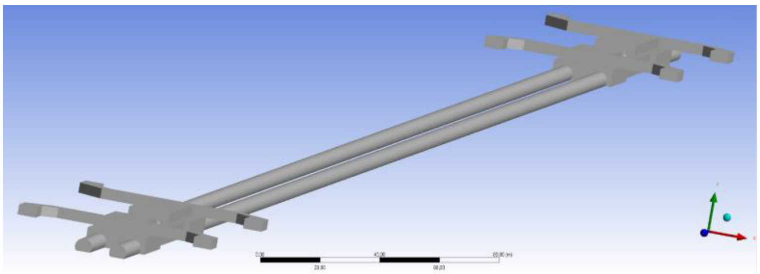


Fig. 1. Geometry of the underground metro rail transport system

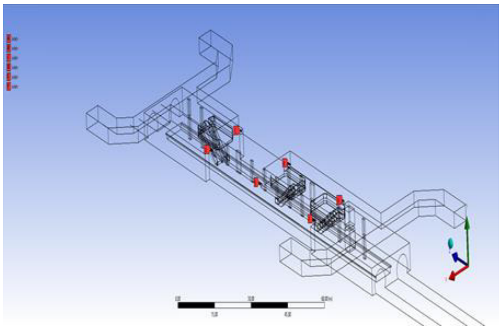


Fig. 2. Details of underground metro rail transport system geometry with ventilation system

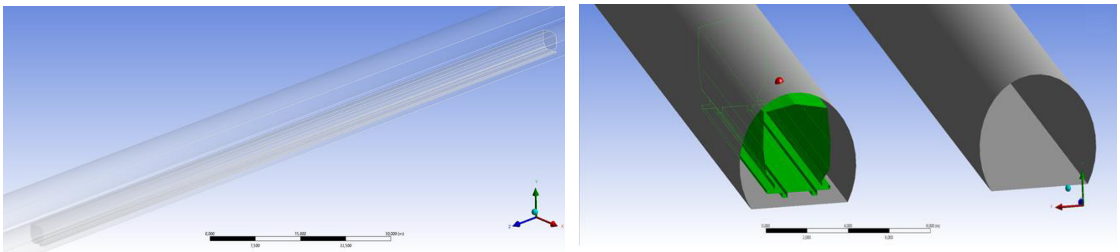
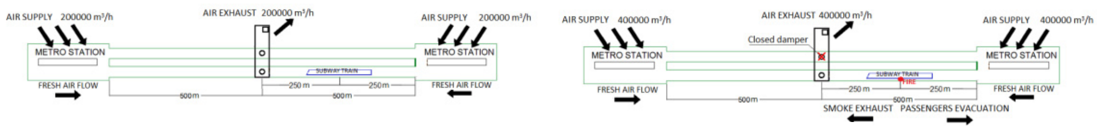


Fig. 3. Geometry of the metro train taken into consideration



a) b)

Fig. 4. Operational scheme of the ventilation system: a) normal functioning; b) emergency functioning

3. CFD modeling

3.1. Computational domain geometry and discretization

The geometry description used in the CFD model has no approximation compared to the real geometry of stations and tunnels. This allows a more realistic CFD approach to model the air flow in the computational domain. Moreover, the geometry of the model includes the real connections of stations to the street level. This leads to realistic pressure couplings between different openings in the computational domain (tunnels, stations entrances).

The discretization of the computational domain is completed using finite volumes, based on the special pre-processor of the CFD software package, Ansys Fluent 15.0. The grid used is an unstructured mesh that contains tetrahedral elements. The principal motivation for this choice is the necessity of a better discretization in regions with strong flow gradients. In the case of an unstructured mesh, this can be realized by refining only the zones where important flow variations take place, without adding excessive cells all over in the computational domain. This unstructured grid advantage has been used in this case to improve the discretization near the inlets and outlets of the ventilation systems and around the train in the tunnel, as well as in the neighborhood of the fire outbreak zone. On the other hand, the grid used (mesh M – Table 1) is based on numerical mesh sensitivity tests carried out in order to reach grid independent solutions. Different meshes were built changing the global minimum allowable size of the discretization element, keeping constant the growth rate (Table 1).

Table 1. Mesh sensitivity tests.

Mesh	Total number of finite volumes	Growth rate	Minimum element size (m)	Maximum element size (m)
M	9,704,173	1.16	0.15	38.40
M1	6,279,267	1.16	0.20	51.20
M2	11,352,744	1.16	0.13	33.28

The quality of mesh M, used finally during the simulations, is presented in Table 2.

Table 2. Characteristics of mesh used for the computations

Skewness	Number of cells	Percent of total number of cells (%)
[0;0.15]	2,633,511	27.14
[0;0.5]	9,355,502	96.41
[0.7;0.95]	23,861	0.25

3.2. Air flow and CO_2 concentrations modeling

The construction of the CFD model is based on CO_2 transport and diffusion in air to estimate the ventilation efficiency. As a result, the fluid is considered a mixture of air and CO_2, with the following main properties: ideal gas, incompressible Newtonian fluid, and no chemical reaction between the species of the mixture. In addition, the diffusion coefficient of CO_2 in air has a constant value (1.6×10^{-5} m^2/s).

The transport and diffusion mechanisms of CO_2 are taking into account in the CFD model using a conservation equation of the CO_2 mass fraction. This equation can be written as a standard convection-diffusion equation:

$$0 = -\nabla \cdot \left(\rho \phi \vec{v}\right) + \nabla \cdot \left(\Gamma \nabla \phi\right) + S \tag{1}$$

where ρ – mixture density; Φ - variable of interest: mass fraction of CO_2, 1 for the continuity, velocity components, temperature, or turbulent parameters; v – mixture velocity; Γ - diffusion coefficient; S – source term.

The first term on the right-hand side of the Eq. (1) stands for purely advection phenomena and it is representing the modification of the CO_2 concentration due to the air flow. The second term on the right-hand side of the Eq. (1) signifies the diffusion term. This term integrates in the CFD model both aspects of diffusion, molecular and turbulent. The molecular diffusion is based on Fick's first law: diffusion flux is proportional to the concentration gradient, using constant diffusion coefficient of CO_2 in air, as mentioned above. On the other hand, the turbulent diffusion of CO_2 is taking into account in a similar way to that of the Reynolds analogy. Consequently, mixture mass turbulent diffusivity is related to the eddy viscosity through the CFD turbulence model used to describe the air flow in the computational domain. Finally, the third term on the right-hand side of the Eq. (1) refers to source terms. The value of the term source in Eq. (1) is given by the CO_2 flow rate which occurs in case of fire.

The CO_2 conservation equation is added to the basic equations describing turbulent confined non-isothermal flows (conservation of mass, momentum, energy, and turbulent quantities) in CFD modelling.

Regarding the air-CO_2 mixture turbulent flow modeling, the simulations are based on the standard k-ε two-equation turbulence model since this approach was widely used for numerous engineering applications with decent results. Furthermore, the k-ε turbulence model was used in similar studies, leading to accurate descriptions of the mean air-smoke flow [12]. On the other hand, comprehensive studies of the flow turbulent structure based on more complex turbulence models did not significantly improve the results as the impact of these detailed turbulent structures on the air-smoke flow is minimum and can be neglected [9].

The near-wall air flow treatment has been taken into account by means of standard wall functions (logarithmic laws) as the flow in our case is under forced convection mode. These logarithmic laws are known to be valid for values of non-dimensional wall distance (y^+) below 30. The minimum value of y^+ throughout all the computational domain, near the tunnel walls, is 46.85 based on the results achieved. This means that the mesh requirements concerning the appropriate use of wall functions are fulfilled in our model.

3.3. Radiative heat fluxes modeling

The radiation modeling within the computational domain is carried out by the discrete ordinates (DO) approach. The DO method used is based on the finite-volume method which solves the radiative transfer equation for a finite number of discrete solid angles [13]. The advantage of this method is that the classic radiative transfer equation is adapted in a transport equation which can be numerically solved as the other equations describing the air-CO_2 mixture flow.

It is worthwhile to mention that we used a minimum angular discretization for the application of the DO method: two control angles for the polar and azimuthal angles to define the vector direction in space.

3.4. Fire modeling

Source terms of heat and CO_2 are used in the numerical model to represent the effects of fire. Consequently, the energy source term is added in the conservation energy equation, while the CO_2 source term is added in the conservation of the CO_2 mass fraction equation. These source terms are introduced under the subway train chassis: two zones of 0.4 x 5.0 m^2 were defined in the lower part of the subway wagon, exactly in the middle (Fig. 5), where time-varying source terms are set according to Fig. 6 for the heat released by the fire and an area of 0.4 x 5.0 m^2, located between the two zones defined for the heat source terms (Fig. 5), is used for the CO_2 fire source term (2.61 kg/s).

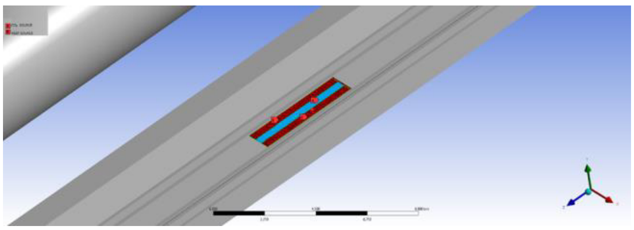


Fig. 5. Heat and CO_2 source terms

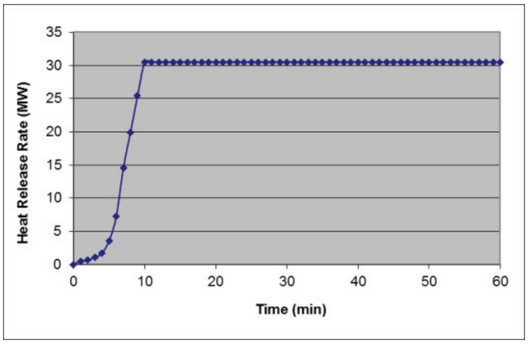


Fig. 6. Heat release rate in time

3.5. Boundary conditions

The following boundary conditions are defined (Fig. 7): inlets ventilation system - set velocity to get the air flow values according to the operating scheme of the ventilation system and turbulent quantities (turbulence kinetic energy and its dissipation rate) computing based on specified values for turbulent intensity (3%) and hydraulic diameter (1.14 m), using the dimensions of the supply inlet (2 x 0.8 m^2); outlets ventilation system and openings in the computational domain (tunnels, stations entrances), pressure equal to atmospheric pressure. This leads to the possibility to calculate the exhaust air flow (or supply air flow, depending on the pressure values) through these openings, based on simulation results.

It is worthwhile to mention that this method leads to pertinent results as the outlets reached the exhaust air flow designed values for the ventilation system (see Fig. 4). In addition, the airflow rates in the computational domain are well balanced. These results express the compulsory condition to sustain the precision of the CFD model both in terms of convergence and air flow.

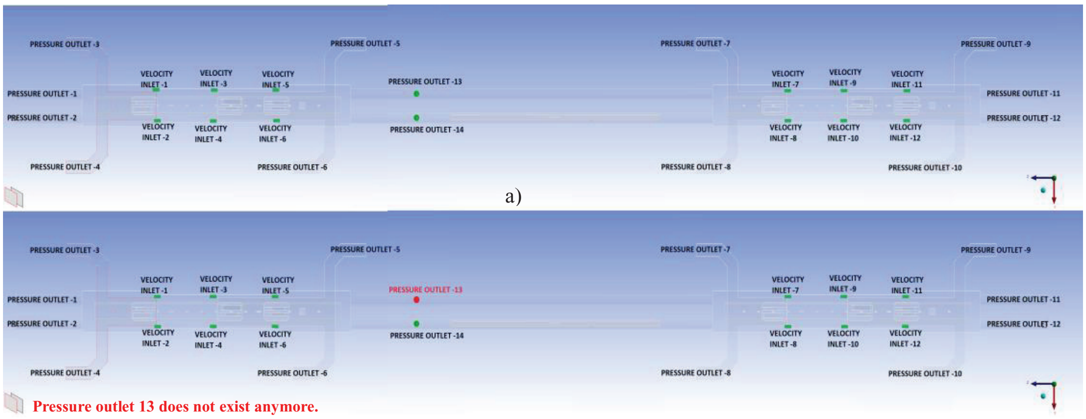


Fig. 7. Boundary conditions: a) normal operating mode; b) emergency (fire) operating mode

3.6. Simulation procedure

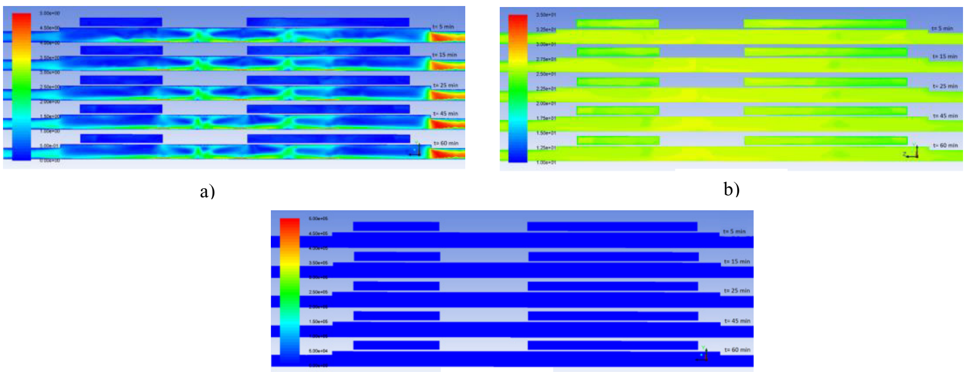
The numerical model is developed based on the CFD software package, Ansys Fluent 15.0, taking into account all the above points. The simulations follow the real operation of the ventilation system in normal and emergency (fire) conditions. In addition, particular attention is paid to the moment when the ventilation system switches from normal operation mode to the emergency mode. In this regard, simulations are first completed in steady-state conditions to describe the normal operation of the ventilation system. After obtaining the steady-state solutions, these are used as initialization values for transient simulations, started at the same time with the fire broke out. This allowed performing the simulations in case of fire based on well-defined initial conditions (air flow, temperature, and CO_2 concentrations) all over the computational domain (stations, tunnels, air ducts, etc.). Transient simulations are carried out for a period of 60 minutes, with an integration time-step of one minute.

4. Results

We present below the results achieved at different time steps in terms of velocities, temperatures and CO_2 concentrations. Fig. 8 shows the data for the farthest metro station from the subway train stopped in the tunnel. The velocity contours reveal that the emergency operation mode of the ventilation system is well established, the values reaching roughly 5 m/s in the tunnel, in comparison with the normal operation mode of the ventilation system where the air velocity does not exceed 2 m/s in the tunnel (this corresponds to the change in air flow rate, from 200,000 m^3/h to 400,000 m^3/h). Fig. 8 shows also that there is no smoke arriving in this station since the levels of temperature and CO_2 concentrations remain in the normal range. It is assumed, on the one hand, that the hot air is equivalent to the presence of smoke [8], and on the other hand, that high concentrations of CO_2 is also synonymous with the existence of smoke [9].

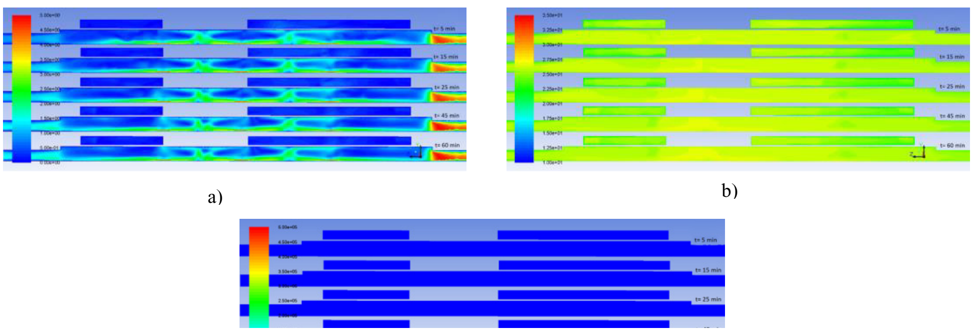
The same applies on the results in Fig. 9 (the nearest station from the subway train stopped in the tunnel). There is no hot air movement or high CO_2 concentrations in this station. This leads to the conclusion that the solution based on mid-tunnel fan plant is efficient. Consequently, this strategy of ventilation is able to provide a safe escape route to the nearest station.

In the region of heat and smoke exhaust (between the train and the location of the mid-tunnel fan plant) – Fig. 10, it can be observed the impact of the subway train stopped in the tunnel on the velocity field (the air velocity is higher on the half-tunnel where there is no subway train), and rapid rise in temperature and CO_2 concentrations (15 minutes: 45ºC and over 30,000 mg/kg CO_2; 25 minutes: 75ºC and around 52,500 mg/kg CO_2; 45 minutes: 120ºC and 60,000 mg/kg CO_2; 60 minutes: 150ºC and approximately 67,500 mg/kg CO_2). It should be noted that these CO_2 concentrations reach dangerous levels that may cause serious health problems.



a) b) c)

Fig. 8. Farthest metro station from the subway train stopped in the tunnel:
a) velocity contours (m/s); b) temperature contours (°C); c) CO_2 concentration contours (mg/kg)



a) b)

Fig. 9. Nearest metro station from the subway train stopped in the tunnel:
a) velocity contours (m/s); b) temperature contours (°C); c) CO_2 concentration contours (mg/kg)

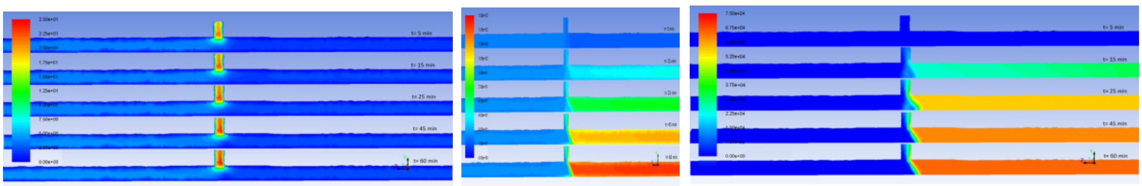
In addition, in the area of the tunnel, in the vicinity of the subway train to exhaust fans (Fig. 11), it can be clearly distinguished the fire expansion in time. The levels of temperature and CO_2 concentrations are extremely high in this region, especially near the fire source (there are spots where values reach 1000ºC and over 500,000 mg/kg CO_2). These temperature peak values are confirmed by experimental data reported in literature [14].

a)

5. Conclusions

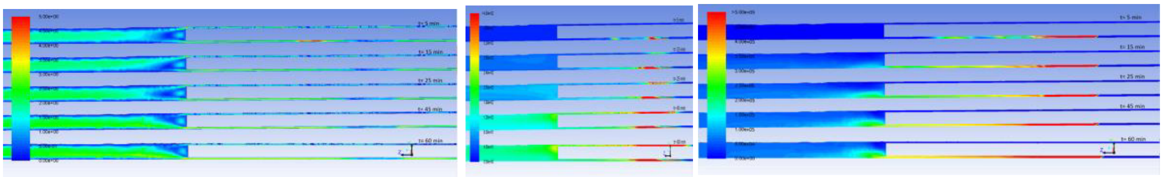
This study has proposed the investigation of the ventilation efficiency for the underground metro rail transport system in the event of a major incident (subway train on fire, stopped in the tunnel). As a result, the ventilation efficacy in this emergency situation is evaluated in this work by the possibility to ensure a safe evacuation path for the passengers in terms of air velocity, air temperature (possibly containing hot gases) and presence of pollutants (represented by CO_2).

The fire is considered as source of heat and CO_2 in the numerical model, by means of a method based on the introduction of source terms in the equation of conservation energy and a special added equation, representing the conservation of the CO_2. This approach allowed obtaining values of velocity, temperature and CO_2 concentrations all over the computational domain (stations and tunnels). The results show that the ventilation strategy taken into consideration leads to the safe evacuation of passengers once they have left the train, as the access to the nearest station is not disturbed by air velocities that may cause difficulty in walking (the predicted values do not exceed 5-6 m/s), high temperatures or dangerous levels of CO_2 to human health (the values of temperatures and CO_2 concentrations remain within normal limits). It must be said that this occurs during all the simulation period (60 minutes).



a) b) c)

Fig. 10. Near the air extraction provided by the tunnel ventilation fan system:
a) velocity contours (m/s); b) temperature contours (°C); c) CO_2 concentration contours (mg/kg)



a) b) c)

Fig. 11. Near the subway train:
a) velocity contours (m/s); b) temperature contours (°C); c) CO_2 concentration contours (mg/kg)

Acknowledgements

This work was supported by a grant of the Romanian Ministry of Education and Research, CNCS-UEFISCDI, project number PN-II-ID-JRP-RO-FR-2012-0071.

References

[1] Murakami H. *Underground: The Tokyo Gas Attack and the Japanese Psyche*. New York: Vintage; 2000.

[2] Lee SR, Nam DG, Moriyama S. A Numerical Study on the Effect of Smoke Control Systems in Subway Station Fires. In: Proceedings of 7th Asia-Oceania Symposium on Fire Science and Technology, Hong Kong, 2007.

[3] Hu L, Wu L, Lu K, Zhang X, Liu S, Qiu Z. Optimization of emergency ventilation mode for a train on fire stopping beside platform of a metro station. *Building Simulation* 2014;**7**:137-146.

[4] Teodosiu C, Ilie V, Teodosiu R. Appropriate CFD Turbulence Model for Improving Indoor Air Quality of Ventilated Spaces. *Mathematical Modelling in Civil Engineering* 2014;**10**:28-42.

[5] Wang F, Wang M. A computational study on effects of fire location on smoke movement in a road tunnel. *Tunnelling and Underground Space Technology* 2015; article in press.

[6] Xi YH, Chow WK, Mao J. Aerodynamics simulation on density jump in a long corridor fire. *Tunnelling and Underground Space Technology* 2015;**50**:23-31.

[7] Zhang B, Zhang J, Lu S, Li C. Buoyancy-driven flow through a ceiling aperture in a corridor: A study on smoke propagation and prevention. *Building Simulation* 2015;**8**:701-709.

[8] Meng N, Hu L, Wu L, Yang L, Zhu S, Chen L, Tang W. Numerical study on the optimization of smoke ventilation mode at the conjunction area between tunnel track and platform in emergency of a train fire at subway station. *Tunnelling and Underground Space Technology* 2014;**40**:151-159.

[9] Chen F, Guo SC, Chuay HY, Chien SW. Smoke Control of Fires in subway Stations. *Theoretical and Computational Fluid Dynamics* 2003;**16**:349-368.

[10] Son BS, Chang HC. Numerical Predictions of Fire Characteristics of Passenger Train Fire in an Underground Subway Tunnel, Depending on Change of Location of Ventilation Facility. *Journal of Korean Institute of Fire Science & Engineering* 2008;**22**:1-8.

[11] Zhou R, Zhang W. Analysis and Optimization of Ventilation Mode of Smoke Control System in Subway Station Fires. *Journal of Theoretical and Applied Information Technology* 2012;**44**:283-289.

[12] Jain S, Shashi, Kumar S. Numerical studies on evaluation of smoke control system of underground metro rail transport system in India having jet injection system: A case study. *Building Simulation* 2011;**4**:205-216.

[13] Siegel R, Howell J. *Thermal radiation heat transfer*. New York: Taylor & Francis; 2002.
[14] Ingason H, Lönnermark A. Recent achievements regarding measuring of time-heat and time-temperature development in tunnels. In: Proceedings of First International Symposium on Safe & Reliable Tunnels. Innovative European Achievements, Prague, Czech Republic, 2004.